VŠB - Technical University of Ostrava

Faculty of Mechanical Engineering

Department of Mechanics

Application Of Super Element Technique In FEM Analysis Of Vessels

Aplikace techniky rozložení oblasti („super-prvky“) při modelování plavidel

Student: Pavel Urubčík

Vedoucí bakalářské práce (supervisor): Doc. Ing. Jiří Podešva, Ph. D.

Ostrava 2009

ANOTACE BAKALÁŘSKÉ PRÁCE

URUBČÍK, P. Aplikace techniky rozložení oblasti („super-prvky“) při modelování plavidel. Ostrava: katedra mechaniky, Fakulta strojní, VŠB - Technická univerzita Ostrava, 2009, 40 s. Bakalářská práce, vedoucí Doc. Ing. Jiří Podešva, Ph. D.

Bakalářská práce se zabývá využitím redukční metody rozkladu na podoblastí v lokálně-globálních konečnoprvkových analýzách. Metoda je užitečná u velmi rozsáhlých úloh. V úvodu je shrnuta teorie problematiky a metoda je následně předvedena na jednoduchém příkladu. V další části je metoda aplikována na skutečném modelu z oblasti lodní techniky. Nejdůležitější poznatky jsou shrnuty a jsou uvedeny zobecněné postupy pro řešení problémů podobné povahy. Na závěr je vyhodnocena funkčnost vypracovaného postupu a jsou podány návrhy pro další rozvoj.

ANNOTATION OF THESIS

URUBČÍK, P. Application Of Super Element Technique In FEM Analysis Of Vessel's. Ostrava: department of mechanics, Department of Mechanics, Faculty of Mechanical Engineering, VŠB - Technical University of Ostrava, 2009, 40 p. , Thesis supervisor Doc. Ing. Jiří Podešva, Ph. D.

The thesis is dealing with application of substructuring, reduction method used in Finite Element Analysis. Substructuring is useful in solution of very large problems. First, theory and the method is introduced. Subsequently, the method is applied on practical offshore equipment problem. General approach for similar problems is described. In conclusion, the developed approach is evaluated and suggestions about further development are given.

Table of Contents

List of Symbols and Abbreviations....2
1.Introduction....3
2.Substructuring....6
2.1.Reduction methods....6
2.1.1.Transformation methods....6
2.1.2.Elimination methods....7
2.2.Mathematics of substructuring....10
2.3.Approach in ANSYS....13
2.3.1.Approach....14
2.3.2.Modelling and its methods....16
2.3.3.Demonstration of substructuring....17
3.Application on the crane slewbearing....20
3.1.Introduction of the problem....20
3.2.Modelling....22
3.2.1.FE model....24
3.3.Results....27
3.3.1.Use pass results....27
3.3.2.Expansion pass results....32
4.Smart guide....36
4.1.Major issues....36
4.2.What to avoid....37
5.Conclusion....38
6.Sources....39

List of Symbols and Abbreviations

Abbreviations

DOF	degree of freedom
MDOF	master degree of freedom
BC	boundary condition
SE	super-element
FE	finite element

Symbol index

K	stiffness matrix
B	damping matrix
M	mass matrix
f	force vector
$\widetilde{\boldsymbol{K}}$	condensed stiffness matrix
$\widetilde{\boldsymbol{B}}$	condensed damping matrix
$\widetilde{\boldsymbol{M}}$	condensed mass matrix
$\widetilde{\boldsymbol{f}}$	force vector of condensed system
$\boldsymbol{q}, \dot{\boldsymbol{q}}, \ddot{\boldsymbol{q}}$	vector of displacement, speed or acceleration in original coordinates
$\boldsymbol{u}, \dot{\boldsymbol{u}}, \ddot{\boldsymbol{u}}$	vector of coefficients of linear combination (transformed displacement and its time derivations)

Subscript index

m	master nodes
s	slave nodes
A, B,δ.	subareas

1. Introduction

Engineering constantly improves its accuracy in means of approaching the reality. For this reason more computer resources are needed since applied knowledge gets more complex.

Computational resources nowadays develop at significant rate. Yet, as engineers take full advantage of the resources, they often come to that even more capacities are needed. Another way to achieve the accuracy are new methods which save the computation resources. Among such methods is also substructuring, method used in finite element analysis.

When were aerospace engineers in 1960s seeking new methods they came out with substructuring. It was used to break down complex systems such as ships or space vehicles. Repetition of parts and parts with natural borders, such as plane and its wings, perfectly fits the substance of substructuring. In the late 1960s substructuring found its way to offshore and shipbuilding industries. It was in Norwegian Offshore Industry where expression “super-element” was used first time.

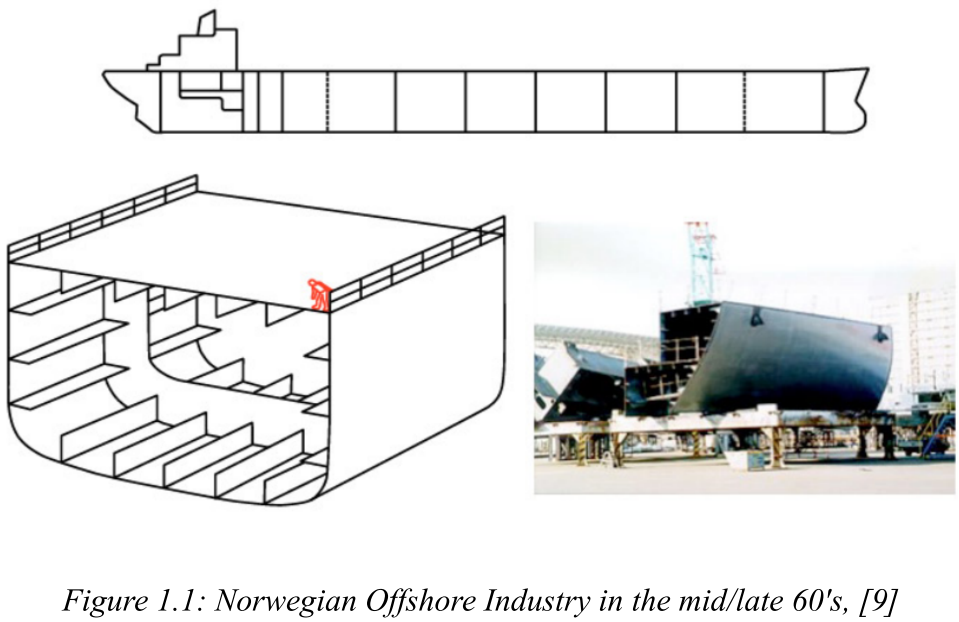

Figure 1.1: Norwegian Offshore Industry in the mid/late 60's, [9]

One of the earliest books about the topic is Przemieniecki's [5]. It fairly describes the beginnings of substructuring. The first application for reduced dynamical models was introduced earlier by Guyan (1965).

Finite Element Method (FEM) solves problem of continuum by dividing it into finite number of items - *elements*. The more elements is used the more accurate is solution. On the other hand the more elements the more computational resources is needed. Elements consist of nodes and degrees of freedom (DOFs. It is the DOF number that determines number of equations to be solved in order to find solution.

Reduction of DOFs becomes a need when appears problem that has to be solved faster (dynamic, optimization) or problem that is so large it cannot even “fit” on computer as whole. Also in some non-linear problems the linear portion can be turned into substructure while only the non-linear portion will be iterated.

It is possible to reduce DOFs without losing accuracy of the results by use of *substructuring,* method examined in this thesis. Group of elements is taken and turned into new element called *super-element (SE)*. The SE DOFs are divided into two groups. One group is used for reduced solution while the rest for full one. Nodes with the first mentioned DOFs are *master nodes* and all the other nodes are *slave nodes*. Advantage is that the full solution of particular SE may be solved only when needed. This computational method also allows to process parts of much more complex structure separately. All this will be further explained in following chapters.

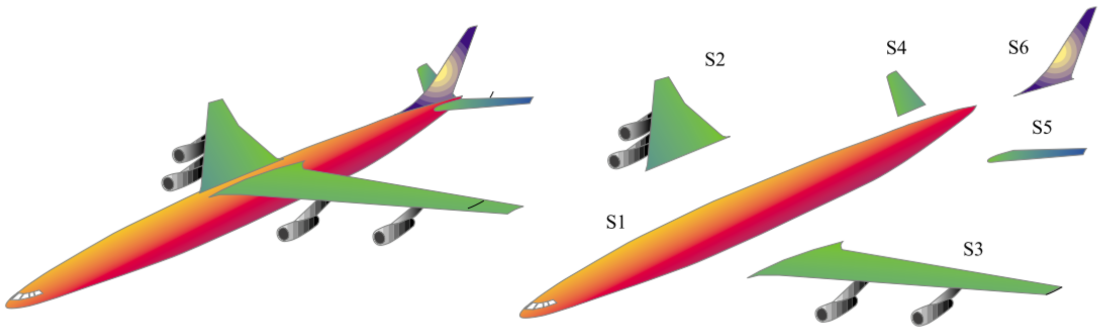

Figure 1.2: Complete airplane broken down into six substructures identified as S1 through S6, [9]

Substructuring can be used in various analyses and in various technology fields. This thesis is focussed on local-global analysis.

Good integration of a heavy crane into a vessel represents an engineering challenge not only for local strength but this often requires consideration of the global strength issue. This means that the local structural analysis has to be combined with global analysis of the whole vessel. Here the *superelement technique* could be a solution.

Substructuring, also called *superelement technique*, will be used in structural analysis of slewbearing of large vessel crane *1800t HLMC.* This topic as well as the documentation is provided by *Huisman – Itrec* company.

The thesis starts with summarizing substructuring and its general application. In next chapter the method is applied on practical problem, the Huisman crane slewbearing.

The aim is to obtain theoretical and practical experience in substructuring, particularly in substructuring of local-global analysis. The piece of knowledge will be applied on the slewbearing problem. Further a guideline for solution of similar problems will be created.

2. Substructuring

2.1. Reduction methods

There are a few reduction methods. They can be divided into two groups. Transformation and elimination methods.

2.1.1. Transformation methods

The key step of this method is that displacement vector (of nodes), load vector and stiffness matrix of a construction are transformed via transformation matrix.

Solution of system of equations (of *n*-th order with *n* unknowns)

$$\boldsymbol{M} \cdot \ddot{\boldsymbol{q}} + \boldsymbol{B} \cdot \dot{\boldsymbol{q}} + \boldsymbol{K} \cdot \boldsymbol{q} = \boldsymbol{f} \qquad (2.1)$$

can be found by linear combination of preselected base vectors. Its number should be $m \ll n$

In matrix form

$$\boldsymbol{q} = \boldsymbol{\Phi} \cdot \boldsymbol{u} \qquad (2.2)$$

where **q** is vector of original nodes displacement, Φ describes base vectors and **u** is vector of coefficients of linear combination. The coefficients are transformed coordinates of fictional nodes into fictional space.

After the transformation the equation (2.1) can be written:

$$\widetilde{\boldsymbol{M}}\cdot\ddot{\boldsymbol{u}}+\widetilde{\boldsymbol{B}}\cdot\dot{\boldsymbol{u}}+\widetilde{\boldsymbol{K}}\cdot\boldsymbol{u}=\widetilde{\boldsymbol{f}} \tag{2.3}$$

It is system of equations of reduced order *m* that represents motion equations in the transformed fictional space.

Modal analysis is typical transformation method. It describes dynamical problem of discrete systems. Modal analysis makes use of natural frequencies. Solution is rather problem of eigenvalues. The system of initial equations is then transformed into system of independent equations making solution easy to find.

2.1.2. Elimination methods

Elimination methods reduce number of degrees of freedom (DOF). Among these methods are static condensation, dynamic condensation and for this thesis most important *superelement technique*.

Static condensation

System that has *n* DOFs has to be divided into two groups. Every DOF is either *Master* (marked with index "*m*") or *slave* ("*s*"). It can be written $n-m=s$. The point is that slave DOFs are condensed only into master DOFs.

Static equation of the system is

$$\begin{bmatrix} K_{mm} & K_{ms} \\ K_{sm} & K_{ss} \end{bmatrix} . \begin{Bmatrix} q_m \\ q_s \end{Bmatrix} = \begin{Bmatrix} f_m \\ f_s \end{Bmatrix} \tag{2.4}$$

written as two separate equations

$$
\begin{aligned}
K_{mm} \cdot q_m + K_{ms} \cdot q_s &= f_m \\
K_{sm} \cdot q_m + K_{ss} \cdot q_s &= f_s
\end{aligned}
\tag{2.5}
$$

when q_s is derived from the second equation

$$
q_s = K_{ss}^{-1} \cdot (f_s - K_{sm} \cdot q_m) \tag{2.6}
$$

then put into the first equation

$$
(K_{mm} - K_{ms} \cdot K_{ss}^{-1} \cdot K_{sm}) \cdot q_m = f_m - K_{ms} \cdot K_{ss}^{-1} \cdot f_s \tag{2.7}
$$

we get *equation for condensed stiffness*

$$
\boldsymbol{\widetilde{K_{mm}} \cdot q_m = \widetilde{f_{mm}}} \tag{2.8}
$$

In the following equation

$$
\begin{aligned}
\widetilde{K_{mm}} &= K_{mm} - K_{ms} \cdot K_{ss}^{-1} \cdot K_{sm} \\
\widetilde{f_m} &= f_m - K_{ms} \cdot K_{ss}^{-1} \cdot f_s
\end{aligned}
\tag{2.9}
$$

$\widetilde{K_{mm}}$ is called the condensed stiffness matrix and $\widetilde{f_m}$ is force vector of the substructure. When used in further operations $\widetilde{K_{mm}}$ and $\widetilde{f_m}$ may be regarded as an stiffness matrix and nodal force vector of an individual element.

Dynamic equation is derived in the same manner

$$\widetilde{\boldsymbol{M}_{mm}} \cdot \ddot{\boldsymbol{q}}_m + \widetilde{\boldsymbol{B}_{mm}} \cdot \dot{\boldsymbol{q}}_m + \widetilde{\boldsymbol{K}_{mm}} \cdot \boldsymbol{q}_m = \widetilde{\boldsymbol{f}_m} \tag{2.10}$$

In this equation

$$\begin{aligned} \widetilde{M_{mm}} &= M_{mm} - K_{ms} \cdot K_{ss}^{-1} \cdot M_{sm} - M_{ms} \cdot K_{ss}^{-1} K_{sm} + K_{ms} K_{ss}^{-1} K_{sm} + K_{ms} \cdot K_{ss}^{-1} \cdot M_{ss} K_{ss}^{-1} K_{sm} \\ \widetilde{B_{mm}} &= B_{mm} - K_{ms} \cdot K_{ss}^{-1} \cdot B_{sm} - B_{ms} \cdot K_{ss}^{-1} K_{sm} + K_{ms} K_{ss}^{-1} K_{sm} + K_{ms} \cdot K_{ss}^{-1} \cdot B_{ss} K_{ss}^{-1} K_{sm} \end{aligned} \tag{2.11}$$

and matrix $\widetilde{K_{mm}}$ and vectors $\widetilde{q_m}, \widetilde{f_{mm}}$ have the same meaning as previously (2.9).

Number of master nodes should be much smaller than number of all nodes because initial matrices **M**, **B, K** have non-zero values only at the diagonal and its surrounding (figure 2.1) while matrices $\widetilde{\boldsymbol{M}_{mm}}, \widetilde{\boldsymbol{B}_{mm}}, \widetilde{\boldsymbol{K}_{mm}}$ are full. Size of $\widetilde{\boldsymbol{M}_{mm}}, \widetilde{\boldsymbol{B}_{mm}}, \widetilde{\boldsymbol{K}_{mm}}$ is set by number of master nodes and its DOFs.

In order to save reasonable amount of resources number of master nodes should be very limited.

$$\begin{bmatrix} \Re & \Re & & & & \\ \Re & \Re & \Re & & & \\ & \Re & \Re & \Re & & \\ & & \Re & \Re & \Re & \\ & & & \Re & \Re & \Re \\ & & & & \Re & \Re \end{bmatrix} \qquad \begin{bmatrix} \Re & \Re & \Re & \Re & \Re & \Re \\ \Re & \Re & \Re & \Re & \Re & \Re \\ \Re & \Re & \Re & \Re & \Re & \Re \\ \Re & \Re & \Re & \Re & \Re & \Re \\ \Re & \Re & \Re & \Re & \Re & \Re \\ \Re & \Re & \Re & \Re & \Re & \Re \end{bmatrix}$$

Figure 2.1: Matrix structure difference between regular (left) and condensed (right)matrices, symbol $\Re$ *for occupied cells*

In substructuring matrices $\widetilde{\boldsymbol{M}_{mm}}, \widetilde{\boldsymbol{B}_{mm}}, \widetilde{\boldsymbol{K}_{mm}}$ are created in first step and the process is called "generation pass".

2.2. Mathematics of substructuring

Substructuring is a process where a group of finite elements is condensed into one element. This element is represented by single matrix and is called *superelement (SE).* Once created SE can be used in analysis as any ordinary element.

A substructure analysis has three steps called *passes*. The first, called *generation pass,* is the process of SE creation. Its mathematics was introduced in the previous chapter (2.1.2) as *static condensation*. The only difference is, that in substructuring, master nodes (its DOFs) are being chosen on particular places of analysed structures.

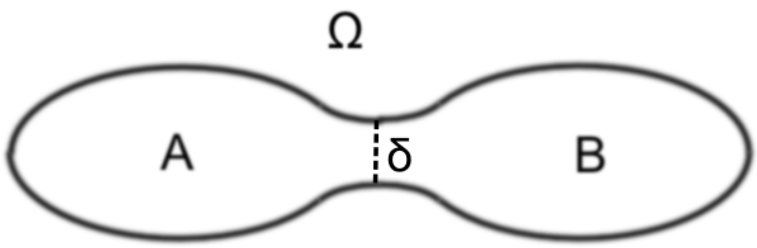

Figure 2.2: Continuum division

In mechanics of continuum, as shown in figure 2.2 , body **Ω** can be divided by border **δ** into two subareas A and B.

General static equation of the problem in matrix form is

$$K \cdot q = f \tag{2.11}$$

with respect to the continuum division

$$\begin{bmatrix} \boldsymbol{K}_{AA} & \boldsymbol{K}_{A\delta} & \boldsymbol{0} \\ \boldsymbol{K}_{\delta A} & \boldsymbol{K}_{\delta\delta} & \boldsymbol{K}_{\delta B} \\ \boldsymbol{0} & \boldsymbol{K}_{B\delta} & \boldsymbol{K}_{BB} \end{bmatrix} \cdot \begin{Bmatrix} \boldsymbol{q}_A \\ \boldsymbol{q}_\delta \\ \boldsymbol{q}_B \end{Bmatrix} = \begin{Bmatrix} \boldsymbol{f}_A \\ \boldsymbol{f}_\delta \\ \boldsymbol{f}_B \end{Bmatrix} \tag{2.12}$$

where K_{AA}, K_{BB} and $K_{\delta\delta}$ are square matrices of subareas A,B and border δ. All the other matrices are rectangular and represent relation of two adjoined areas (or area and border) indicated in the index. Vectors q_A, q_B and $q_{\delta\delta}$ represent displacement of the areas (or potentially nodes inside). Vectors f_A, f_B and $f_{\delta\delta}$ represent forces within the areas.

The matrix equation (2.12) written as system of equations

$$\begin{aligned} K_{AA} \cdot q_A + K_{A\delta} \cdot q_\delta &= f_A \\ K_{\delta A} \cdot q_A + K_{\delta\delta} \cdot q_\delta + K_{\delta B} \cdot q_B &= f_\delta \\ K_{B\delta} \cdot q_\delta + K_{BB} \cdot q_B &= f_B \end{aligned} \tag{2.13}$$

from which q_δ can be derived

$$(-K_{\delta A}\cdot K_{AA}^{-1}\cdot K_{A\delta}+K_{\delta\delta}-K_{\delta B}\cdot K_{BB}^{-1}\cdot K_{B\delta})\cdot q_\delta=f_\delta-K_{\delta A}\cdot K_{AA}^{-1}\cdot f_A-K_{\delta B}\cdot K_{BB}^{-1}\cdot f_B \tag{2.14}$$

When the following substitution is used

$$\begin{aligned}\widetilde{K_\delta}&=-K_{\delta A}\cdot K_{AA}^{-1}\cdot K_{A\delta}+K_{\delta\delta}-K_{\delta B}\cdot K_{BB}^{-1}\cdot K_{B\delta}\\ \widetilde{f}_\delta&=f_\delta-K_{\delta A}\cdot K_{AA}^{-1}\cdot f_A-K_{\delta B}\cdot K_{BB}^{-1}\cdot f_B\end{aligned} \tag{2.15}$$

we get reduced form

$$\widetilde{\boldsymbol{K}_\delta}\cdot\boldsymbol{q}_\delta=\widetilde{\boldsymbol{f}_\delta} \tag{2.16}$$

Solution of (2.16) has to be found in order to get displacement vector q_δ. This is the second step in substructuring analysis called *use pass*.

Suppose areas A and B are elements and border δ between them is area with nodes that are shared by both elements. Then problem in figure 2.1 could be regarded as regular finite element analysis, only with very limited number of elements.

In order to obtain solution inside the substructures A and B, the following equations (2.17 and 2.18) have to be solved. That is the last step of substructuring analysis called *expansion pass*.

$$K_{AA}\cdot q_A=\widetilde{f}_A \tag{2.17}$$

In this equation

$$\widetilde{f}_A = f_A - K_{A\delta} \cdot q_\delta \tag{2.18}$$

This chapter summarized substructuring mathematics while the next chapter approaches the problem in practical matter.

2.3. Approach in ANSYS

So far substructuring was explained in general concept. From here onwards the method will be applied in FEA software package *ANSYS [1]*. Substructuring is available in the ANSYS Multiphysics, the ANSYS Mechanical, and the ANSYS Structural products. Some approaches may be still valid out of the ANSYS environment though.

Before the approach of substructuring will be introduced lets define the term, because super-element technique or *substructuring* is often confused with term *submodeling*.

Submodeling is process of taking a coarsely-meshed solution, remodelling its portion, refining the mesh and applying the results of the previous run as boundary conditions for the refined model.

On the other hand, ***substructuring*** is method that splits structure into a series of smaller structures – the substructures. The internal freedoms of a substructure are eliminated and the SE matrix for the substructure is created (*generation pass*). The complete problem is solved by only assembling the freedoms on the common boundaries between the substructures (*use pass*) using the SE matrix. If solution of substructure internal DOFs is needed then *expansion pass* is applied (see chapter 2.3.1).

2.3.1. Approach

Using substructuring

The whole process of substructuring is divided into three steps called “passes”. These are:

1. Generation pass – step where group of elements is condensed into SE
2. Use pass – step similar to any regular analysis but previously created SEs is used and combined with ordinary elements
3. Expansion pass – SE used in *use pass* model is solved via use pass results

Generation pass

In this step group of ordinary elements is condensed into single SE. This is done by identification of *master degrees of freedom* (MDOFs). These can be defined by software or user. MDOFs should be picked in considerable regions of analysed structure. These are even areas where SE is connected to the rest of structure (this might be another SE) or nodes where boundary conditions (BC) will be applied. Other than nodes with MDOFs inside SE are called *slave nodes*. Slave nodes are neglected during *use pass*.

Number of MDOFs should be much smaller than number of all nodes in SE since it has significant impact on solution time. SE stiffness matrix is full unlike matrix of regular model.

There is limitation about element types available in generation pass– only linear elements can be used. Mathematically, *generation pass* is represented by equation 2.9.

Use pass

Use pass is where SE is used as a part of model. Other elements, even non-linear, may be connected to the SE. Solution of *use pass* consists of the reduced solution of SEs and complete solution for non-superelements.

The step can involve all ANSYS analysis types except FLOTRAN and explicit dynamics analysis. Equation 2.8 describes the process.

Expansion pass

In *expansion pass,* the reduced solution from *use pass* is taken and solution in all DOFs (on *slave nodes*) is calculated inside SE. The process starts with loading SE model created in *Generation Pass* and pointing the *Use pass* results. The internal DOFs on slave nodes are solved as expressed in equation 2.6.

Where is good to use substructuring

A key feature of substructuring is to cut down computer time. It also allows to solve large problems with limited computer resources. Substructuring would be already implemented in software as default if it would accelerate any analysis. There are issues to consider before using the method.

The first thing to consider is whether are regions of the model relatively independent. Regions share only a few nodes with one another but yet the regions are needed in their fully geometric detail and complexity for proper system behaviour. The approach is similar when considering cases where the non-linearity is localized. For instance small regions of contact among many relatively large rigid bodies.

Typical candidates are non-linear analyses and analyses of structures containing repeated geometrical patterns. Also cases where only small but detail part of large system is of interest still full model load must be embodied. The latter is called *local-global analysis* and is of particular interest in this thesis.

Substructuring is widely used in aeronautics (figure 1.2) and shipbuilding (figures 1.1 and 2.2).

2.3.2. Modelling and its methods

Different modelling approaches are needed to solve problems of different size and complexity.

Bottom-up substructuring

Method used for very large models which could not "fit" on the computer. First, each SE is separately generated in *Generation Pass*. Afterwards SEs are assembled in *the Use Pass*.

Top-down substructuring

This method is suitable for substructuring of smaller models, global project geometry controls or for isolated component analysis. The latest could be, for instance, substructuring of the linear portion of non-linear models that would fit on the computer. Compared to *Bottom-up* this method allows to assemble results for multiple SEs in postprocessing. There is no need to change database file, although results for different SE appear in different result set.

Nesting

When SE is generated, previously generated SE may be used as any other element that is allowed in *Generation Pass*. The generated SE is called *nested super-element*. For every level additional *expansion pass* is needed. Nesting can be very powerful tool in substructure analysis.

Automatically Generated SE

Both *top-down* and *bottom-up* substructuring need repetition of a set of /SOLU commands for each SE to be generated. SE can be generated automatically by this method. It simplifies the generating process of SE and efficiently breaks a larger model into smaller models, for example, to be used in a non-linear analysis. Method is effective when one need automatically and quickly create superelements (.SUB files), and define master DOFs on interfaces between each superelement.

2.3.3. Demonstration of substructuring

Substructuring analysis will be demonstrated on simple ladder construction. Aim is stress analysis in the ladder. One may presume that whole model is too large for common analysis and therefore needs substructuring.

Generation pass

Super-element selection: crossbar section and its proportion *(*figure *2.3)* is regarded as sufficiently small, still repeated pattern, therefore selected as super-element portion.

Master nodes selection: master nodes must be selected not only on interfaces between two SEs but also with regard to applied *boundary conditions* (BC). In this case vertical lineload in center of the crossbar will be applied.

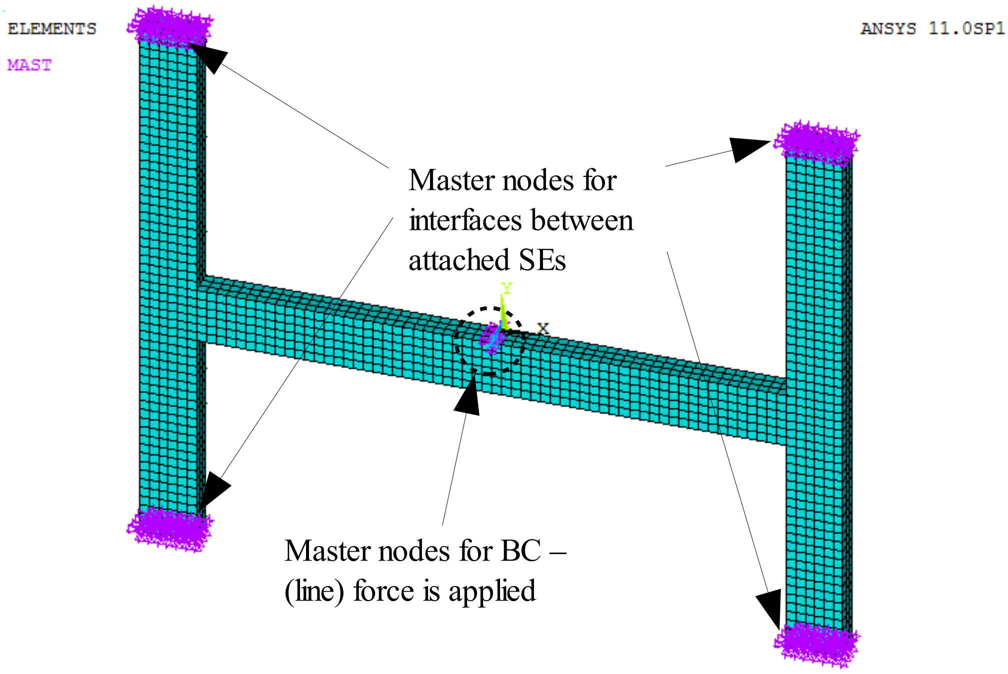

Figure 2.3: generation pass - crossbar portion of the model used as super-element with selected master nodes.

Use pass

Created SE is repeatedly used with geometry offset in regular ANSYS analysis in order to make whole ladder model (figure 2.4). BCs are applied (load force and displacement constraint). SEs are consequently connected via coupling. Regular analysis is performed.

Expansion pass

SE model created in generation pass is loaded. Further, ladder portion, that is of interest, is pointed and *use pass* result file is set. Analysis for all nodes within the SE is performed and postprocessing follows.

Expansion pass results for identical SEs parent (derived from same .tri file in gen pass) are saved in the same set if needed.

Note that only result for one SE may be postprocessed at once. Expansion pass result in figure 2.5 was created by graphical assembling of all crossbar portion results.

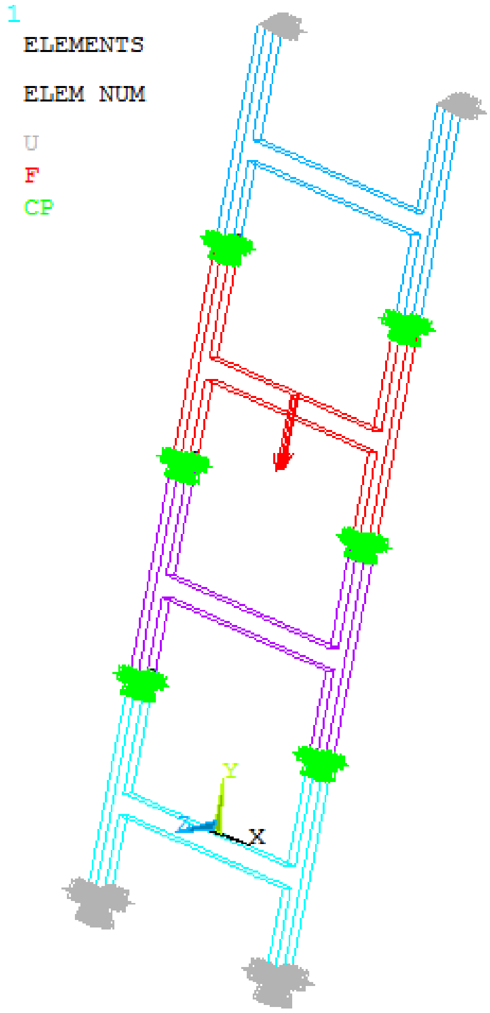

Figure 2.4: multiply used SE in Use Pass with coupled master nodes (CP), applied force (F) and displacement (U) boundary conditions

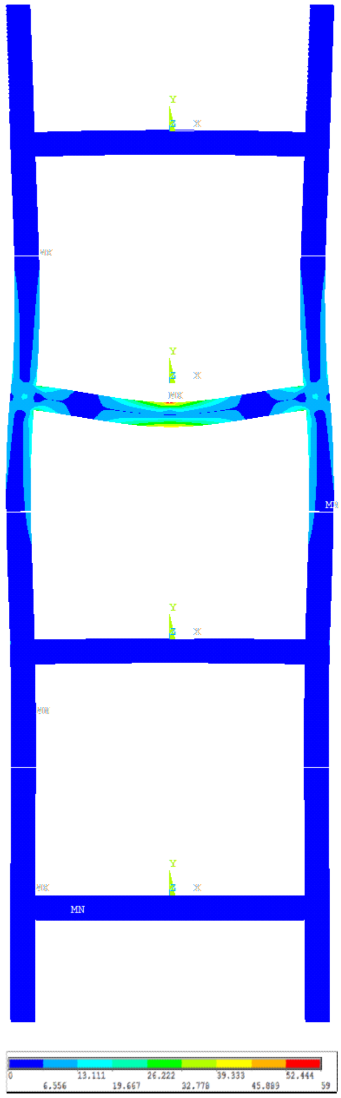

Figure 2.5: Expansion pass results merged into one image (out of ANSYS)

3. Application on the crane slewbearing

3.1. Introduction of the problem

Application of substructuring will be performed on Huisman mastcrane masthead.

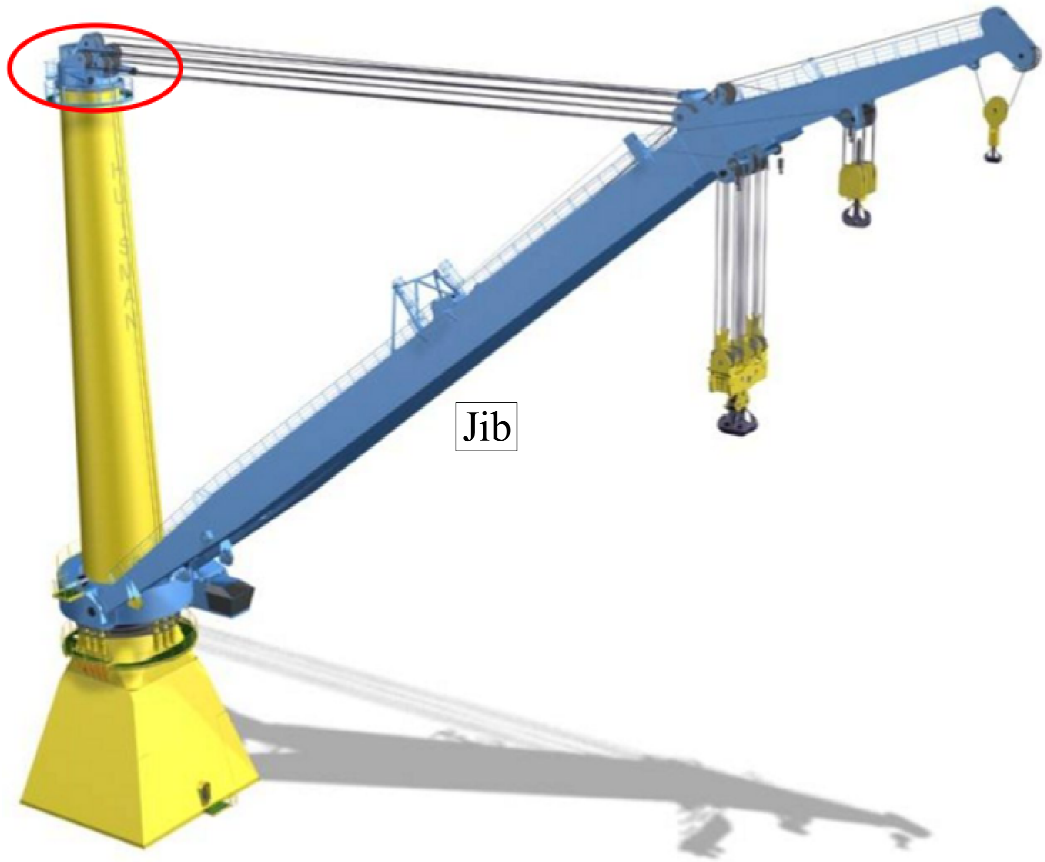

Figure 3.1: Typical Huisman mastcrane, with all rotating parts in blue and all static parts in yellow, [6]

The bearing characteristics of interest:

- Bolt loads due to pretension
- Bolt loads as a function of external loading
- Stresses in the flanges
- Load distribution over the bearing circumference along each raceway
- Stress distribution in the bearing
- Stresses below the rolling elements of each raceway (Herzian stresses)
- Displacements in the bearing

Bolt info

Yield stress	[MPa]	900
Diameter	[mm]	73,5
v - Pretension level	[-]	90 %
Cross section area	[mm^2]	4245
α_A - pretension guarantee factor	[-]	1,2
Moment of inertia	[mm^4]	1,4E+06

Bearing and flange yield stress

Yield stress ring material	[MPa]	700
Yield stress flange material	[MPa]	690

Bolt pretension

Required pretension force (for jack) $F_v = \sigma_{bolt\_yield} * v * A_s$ [kN]

Minimum guaranteed pretension level $\mathbf{F_{ax} = F_v / \alpha_A}$ **[kN]**

$$F_{ax} = F_v / \alpha_A = \sigma_{bolt\,yield} \cdot v \cdot A_s / \alpha_A$$
$$F_{ax} = 900 \cdot 0{,}9 \cdot 4245 / 1{,}2$$
$$F_{ax} = 2865 [kN]$$

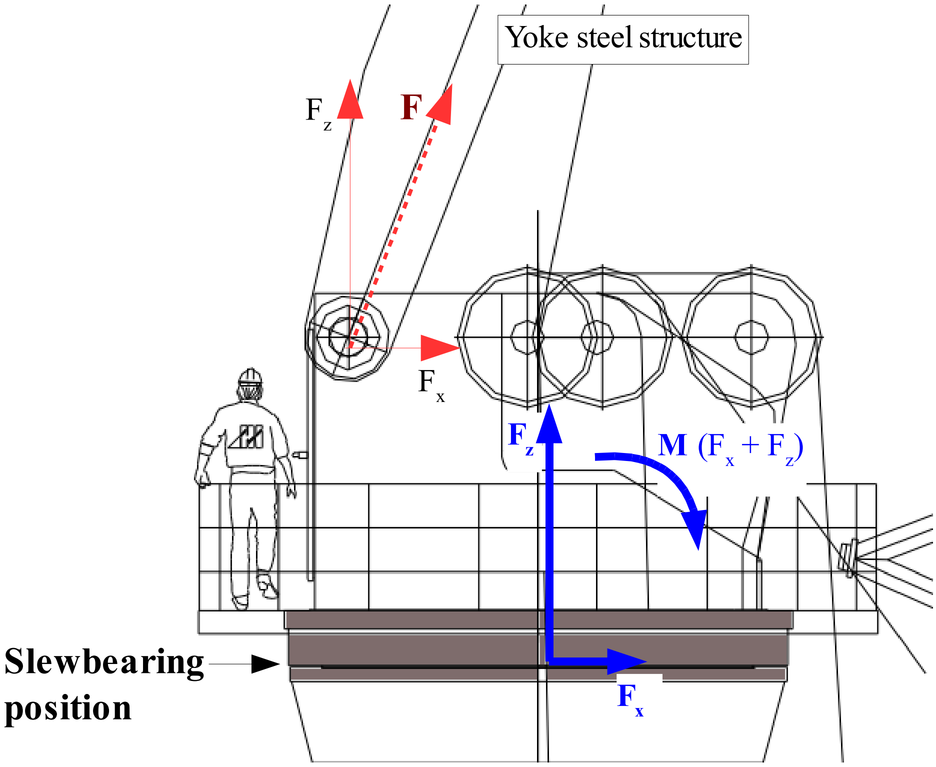

Figure 3.2: detail of figure 3.1;slewbearing and masthead, applied forces (red) and force decomposition (blue)

3.2. Modelling

It is assumed that there are 96 holes in both bearing parts and the same number of rolling elements between them.

Defining super-elements

It is needed to determine exactly which parts of the model will be turned into SEs and which will be treated as non-SE part of model. Non-linear elements,such as LINK10, simulating the slewbearing rolling elements, are forbidden in *generation pass*. Therefore there has to be at least two SEs for the slewbearing (figure 3.3). Flanges were also chosen to be parts of SEs. The rest of model (made mostly of shell elements) will be the non-SE part.

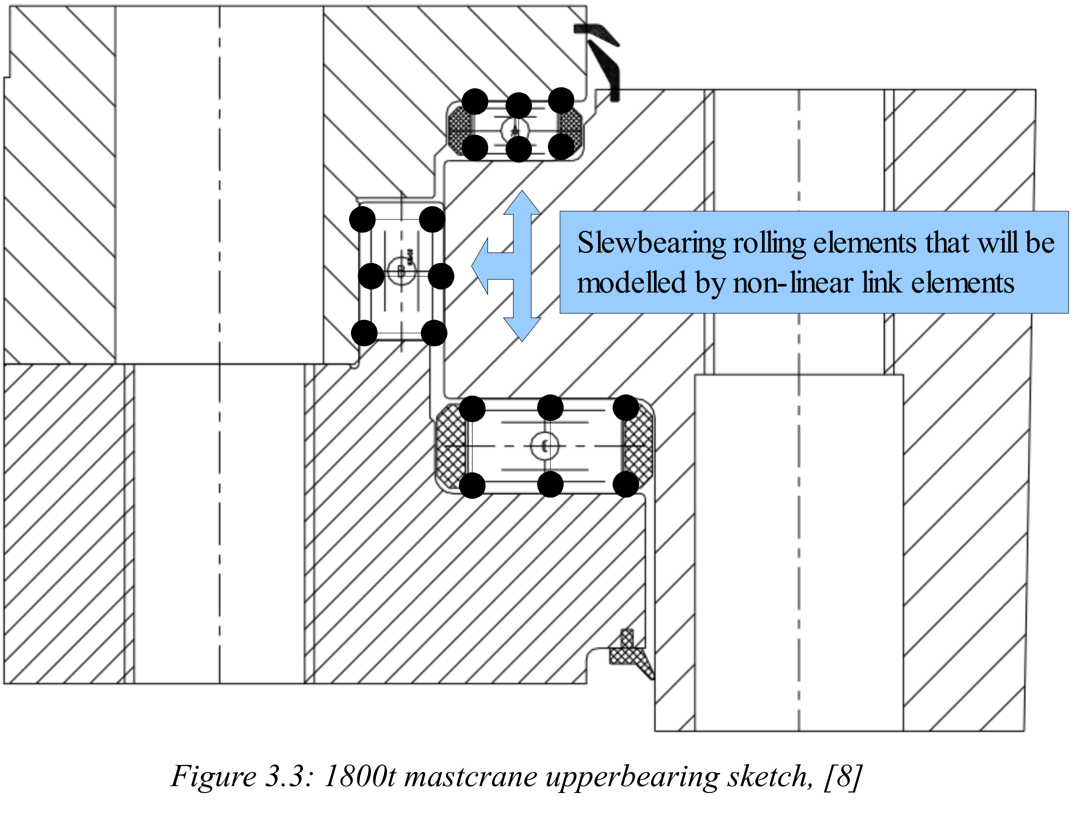

Figure 3.3: 1800t mastcrane upperbearing sketch, [8]

Regard to nature of the problem technique *bottom-up* is used (chapter 2.3.2.). Both parts of the slewbearing are divided by 90 degrees along the circumference leading into 8 SEs (figures 3.4 and 3.5).

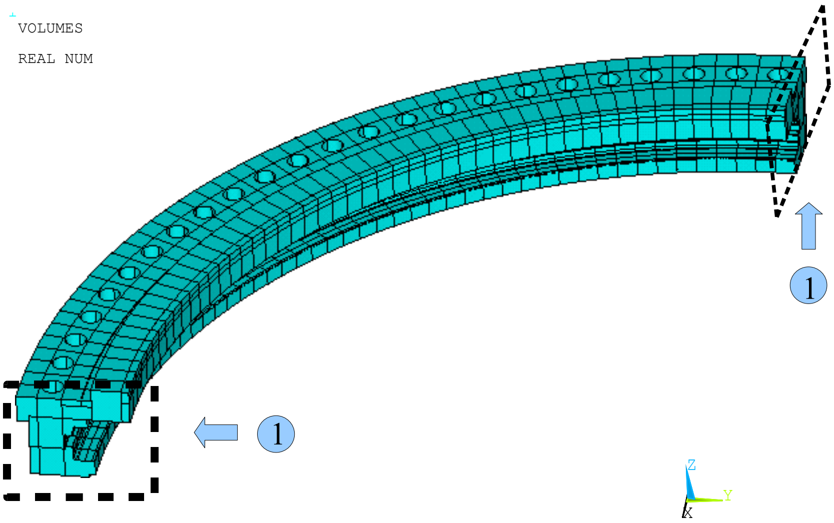

Figure 3.4: Upper part of the slewbearing - 90° SE segment

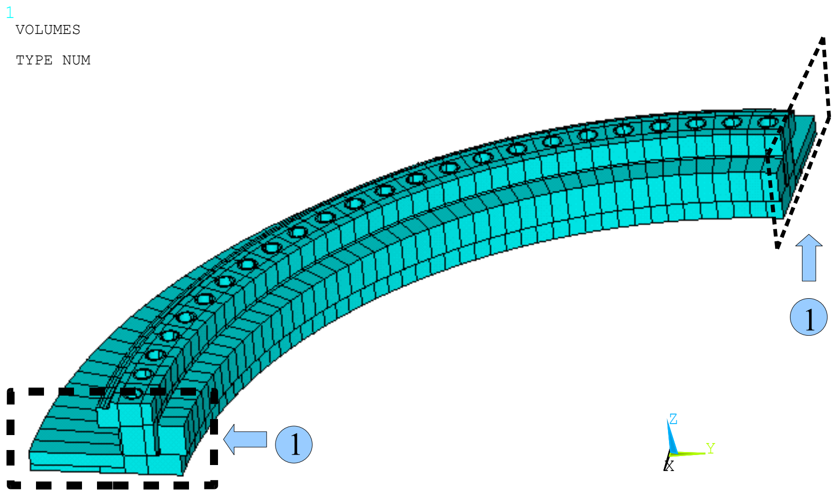

Figure 3.5: Lower part of the slewbearing - 90° SE segment

SE implementation in *use pass*

For use pass implementation it is vital to identify places where SEs interact with surroundings. There are 4 types of such places. It is where **master nodes** will be **selected**.

1. SEs within the same slewbearing part (upper or bottom) are connected with each other (1 in figures 3.4 and 3.5). Depending on mesh fineness the described interface leads into significant number of master DOFs.

2. Rolling elements of the bearing are simulated by non-linear link10 elements. Three links for every rolling element were chosen (figure 3.3).

3. Interface between SE and the non-SE part of model. Since solid has 3 DOFs and shells elements have 6 DOFs one must ensure that all DOFs are transferred.
4. Pretensioned bolts will not be part of the SE, therefore nodes on both ends must be selected as masters.

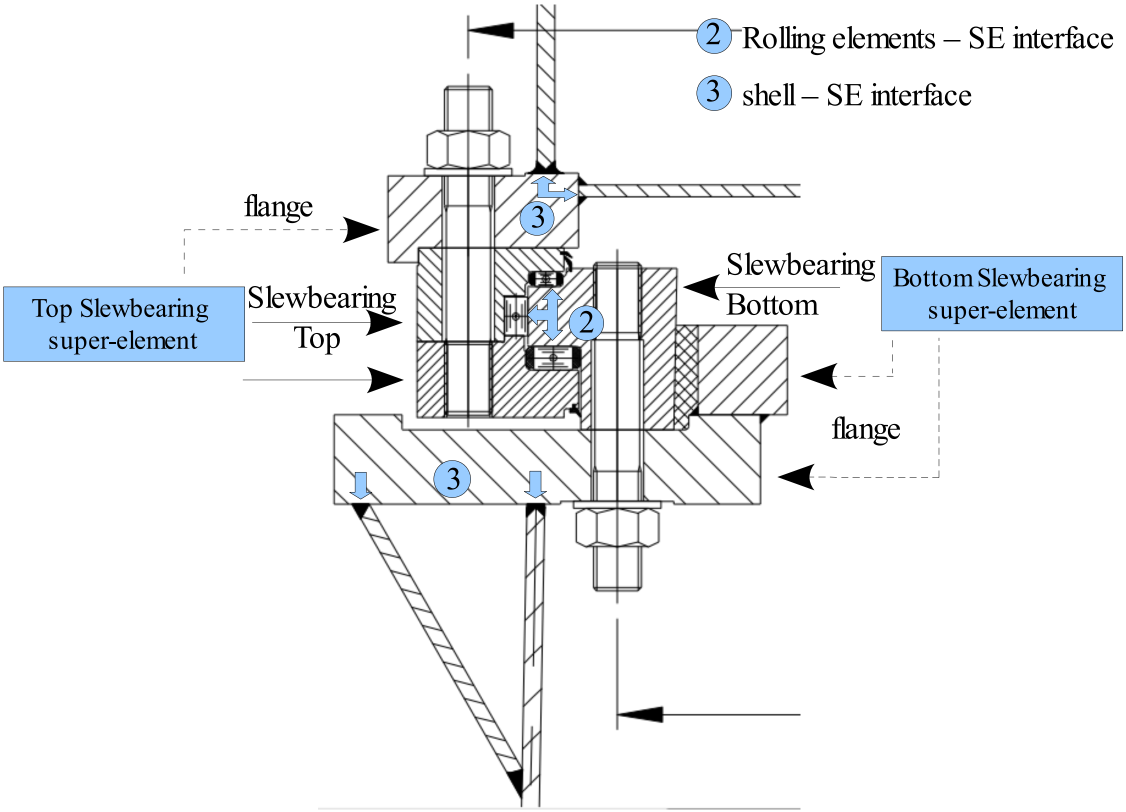

Figure 3.6: general design of the upperbearing, flanges and surroundings

Pretensioned bolts

Pretensioned bolts (figure 3.6) are simulated by LINK8 and the pretension is set as *strain* attribute. The strain is derived from F_{ax} (bolt axial force).

$$\frac{\Delta l}{l}=\frac{F_{ax}}{E\cdot S}=\frac{2865\cdot 10^{3}}{2{,}1\cdot 10^{5}\cdot 4245}=3{,}2139\cdot 10^{-3}$$

3.2.1. FE model

Meshing technique

One hole segment was first created in order to model the 90 degree SE. The segment was fully meshed by the sweep method. Also solid element *without* midnodes were chosen. The mentioned choices reduces number of master nodes which is desirable in order to keep computation time low.

Master nodes selection

The complete 90 degree SE part is created by copying the hole segment. Afterwards master nodes are selected as explained earlier and pointed in figures 3.3 - 3.6.

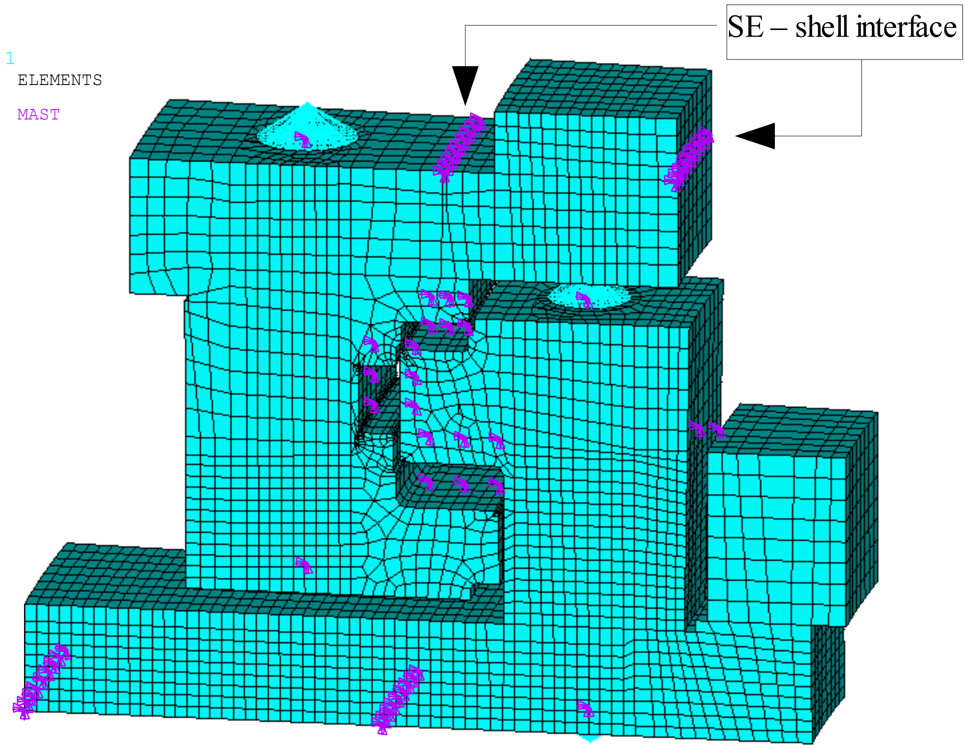

Figure 3.7: slewbearing - one hole segment with selected master nodes, model realization that was shown in figure 3.6

SE – shell joining

Since SE is made of solid elements and non-SE portion of model is made of shell elements it is needed to ensure that all degrees of freedom, including rotations, will be transferred. Shell element, unlike solid element, has rotation DOFs. Master nodes on SE – shell interface must have rotation DOFs too.

This is done by embedding elements with rotation freedoms into the SE. The realization may be seen in figure 3.8. Light blue lines indicate embedded beam elements that provide rotation DOF for master nodes. The beams may be part of SE or may be embedded later in *use pass*. If elements with additional rotation DOF are embed in use pass (used in this analysis) one must take it into account when selecting master nodes in *generation pass* because beams could not be added in use pass if there would not be any extra master nodes on the SE.

The beam element stiffness must be set to small number for negligible impact on analysis. Note that if the parameter is too small it may lead in analysis failure because of great difference between maximal and minimal pivot in system of equations.

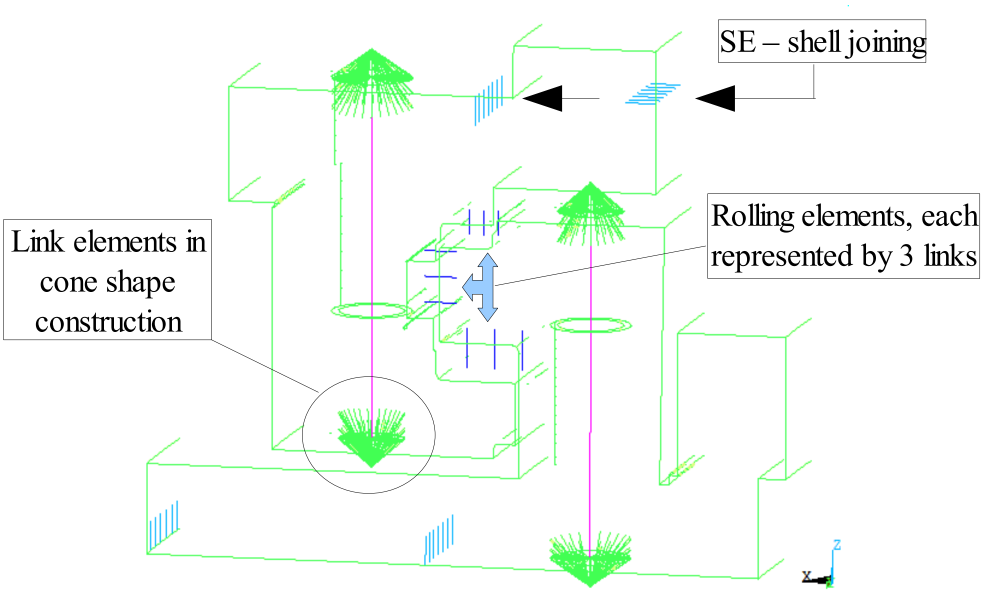

Figure 3.8: SE slice view in use pass model; rolling elements, pretensioned bolt and shell interfaces are visible

The bolt implementation

The bolt is added to the model in *use pass*. Connection between SE and the bolt is reduced to two nodes. One at each end of the bolt. For this is needed to simulate the connection. About three rows of nodes were seleceted along bolt hole circumference at each bolt ending and were connected with additional nodes in the center. This creates cone shaped construction where node in the centre is selected as master node. Situation is well shown in figure 3.8.

Use pass model

Below is shown implementation of SE in use pass model.

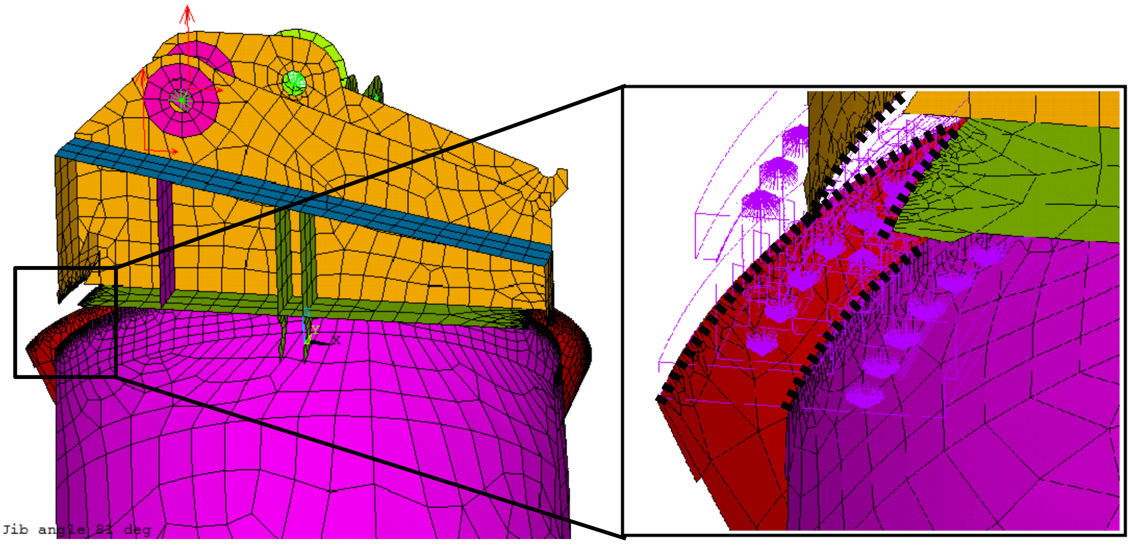

Figure 3.9: Use pass – masthead model, non-SE portion of model and detail of SE - shell interface (on the right)

3.3. Results

All presented results are for Jib angle 81 degrees (figure 3.1) and force F $2x10^7$ N.

3.3.1. Use pass results

Load in rolling elements

Rolling elements are the only elements that transfer load between upper SE to bottom SE. The load in all rolling elements along circumference is shown in *figures 3.10 - 3.12* expressed as axial force in link elements. There are 3 non-linear links elements (LINK10) for each rolling element in the bearing.

Note: Due to modelling issue, in order to get real load value for one rolling element *it is needed to multiply value in figures 3.10 - 3.12 by 6.*

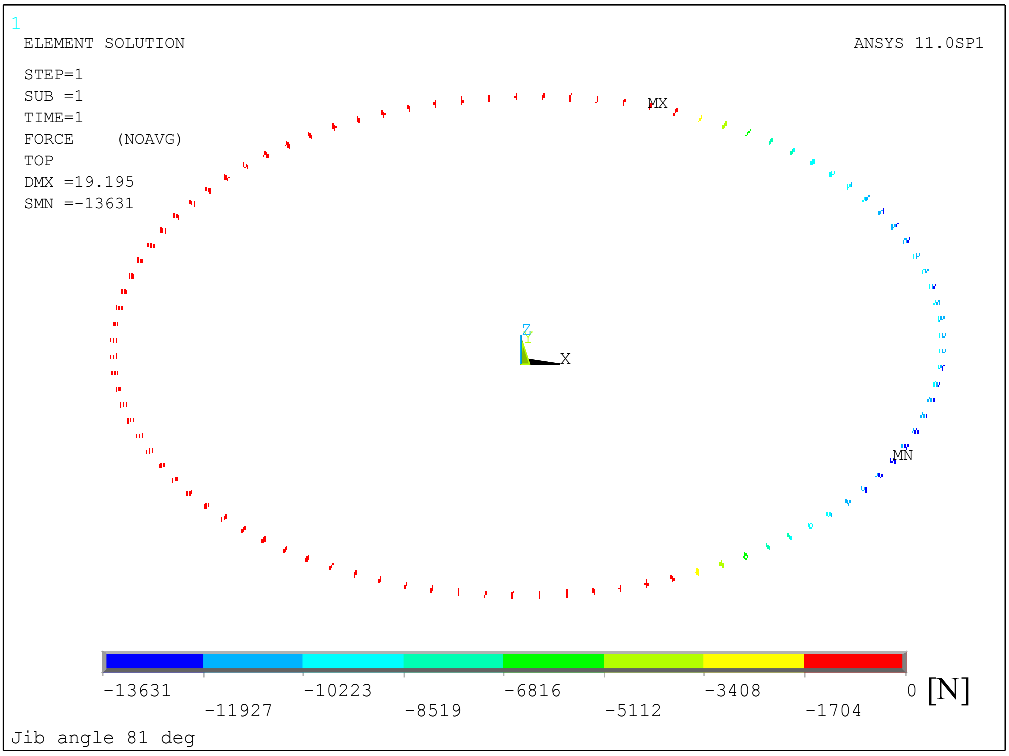

Figure 3.10: Load in upper rolling elements

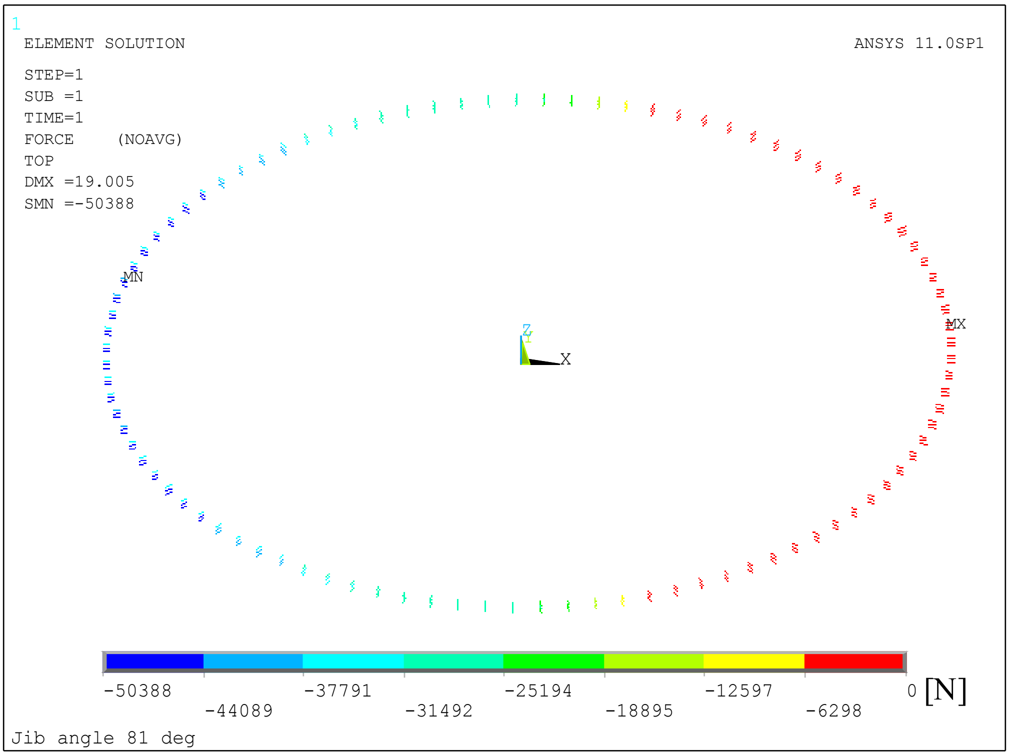

Figure 3.11: Load in radial rolling elements

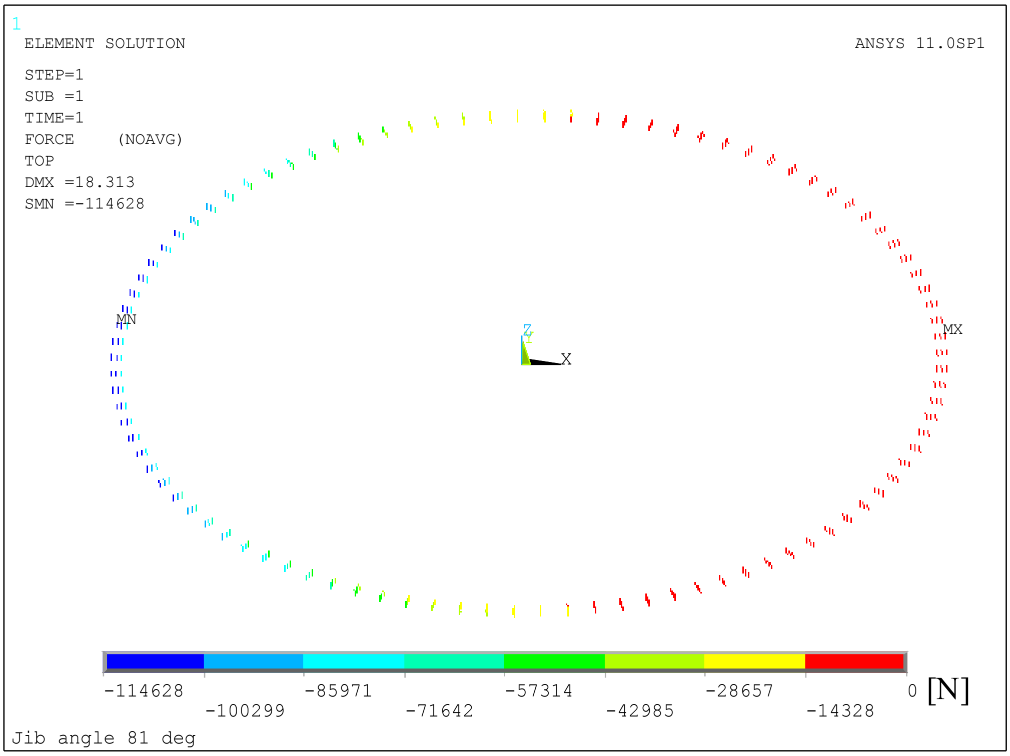

Figure 3.12: Load in bottom rolling elements

Lineload

That is how load in rolling elements spreads over the bearing circumference. Lineload in all three raceways is roughly illustrated in figure 3.13. One may compare with results from design report [7] in figure 3.14.

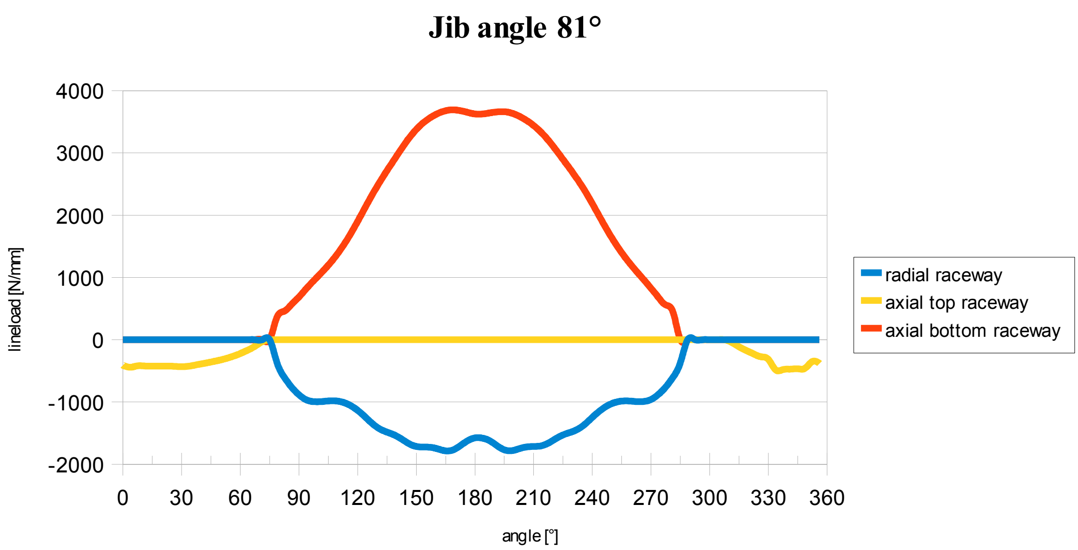

Figure 3.13: Lineload summary

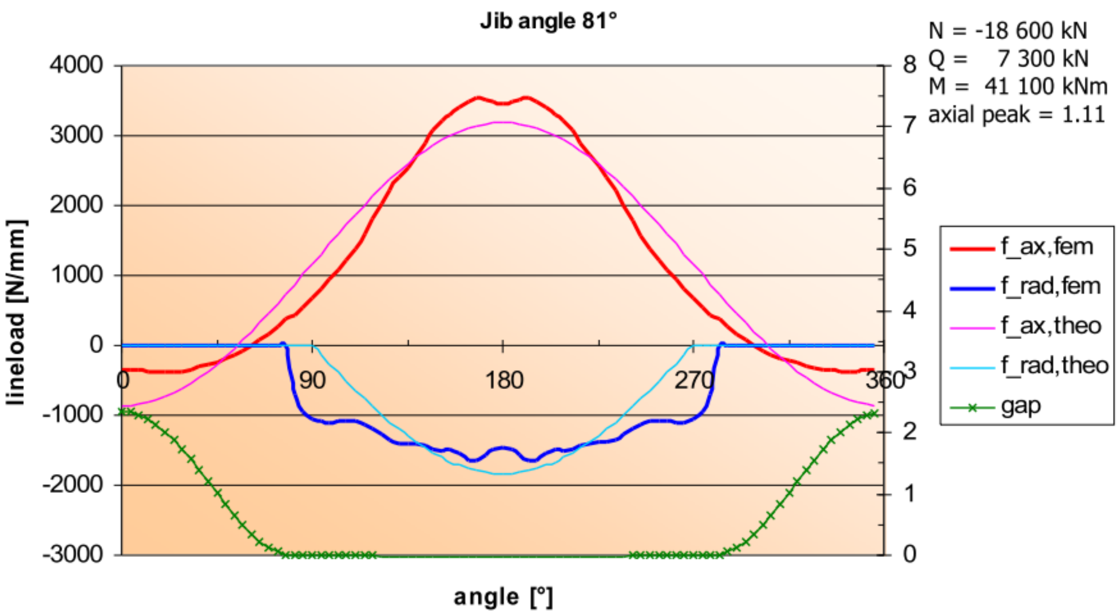

Figure 3.14: Lineload and gap results from [7]

Elastic strain in rolling elements

Elastic strain in figures 3.15 - 3.17 illustrate places with stressed rolling elements (the negative value). At the same time it shows where gaps between bearing and rolling elements appear (positive strain value).

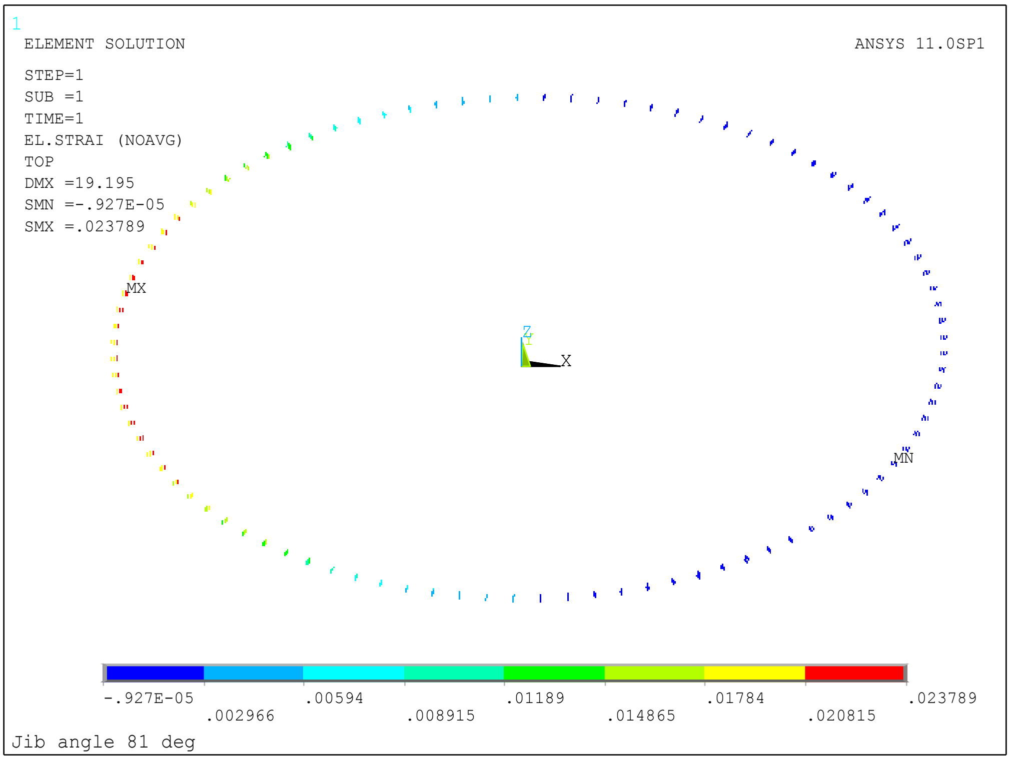

Figure 3.15: Elastic strain in upper rolling elements

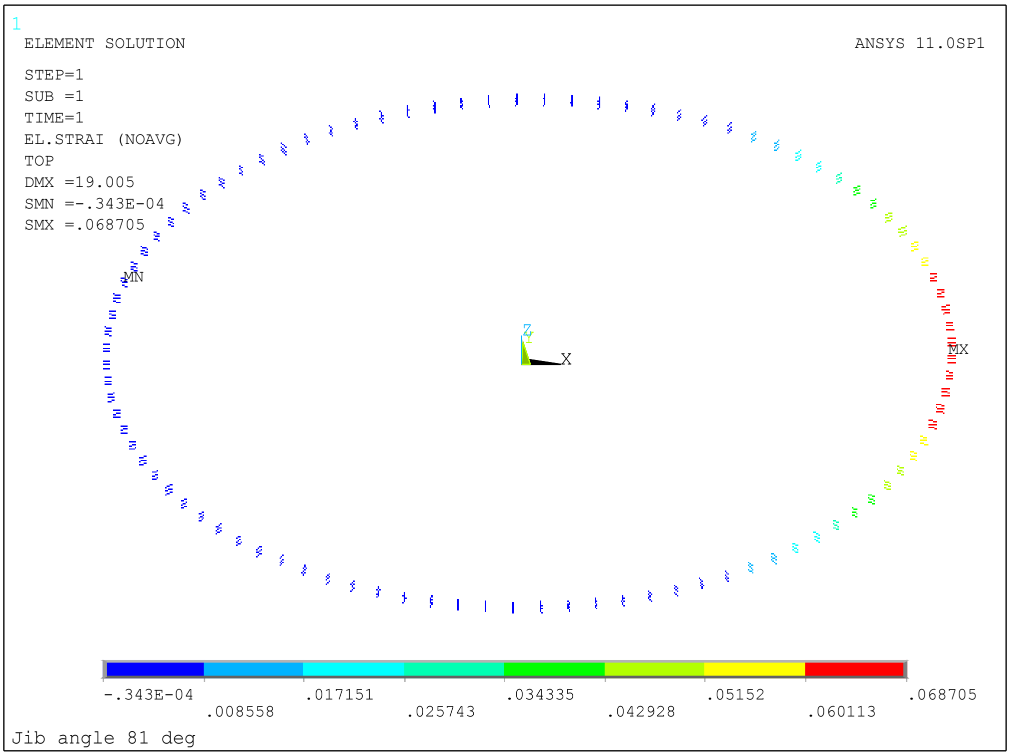

Figure 3.16: Elastic strain in radial rolling elements

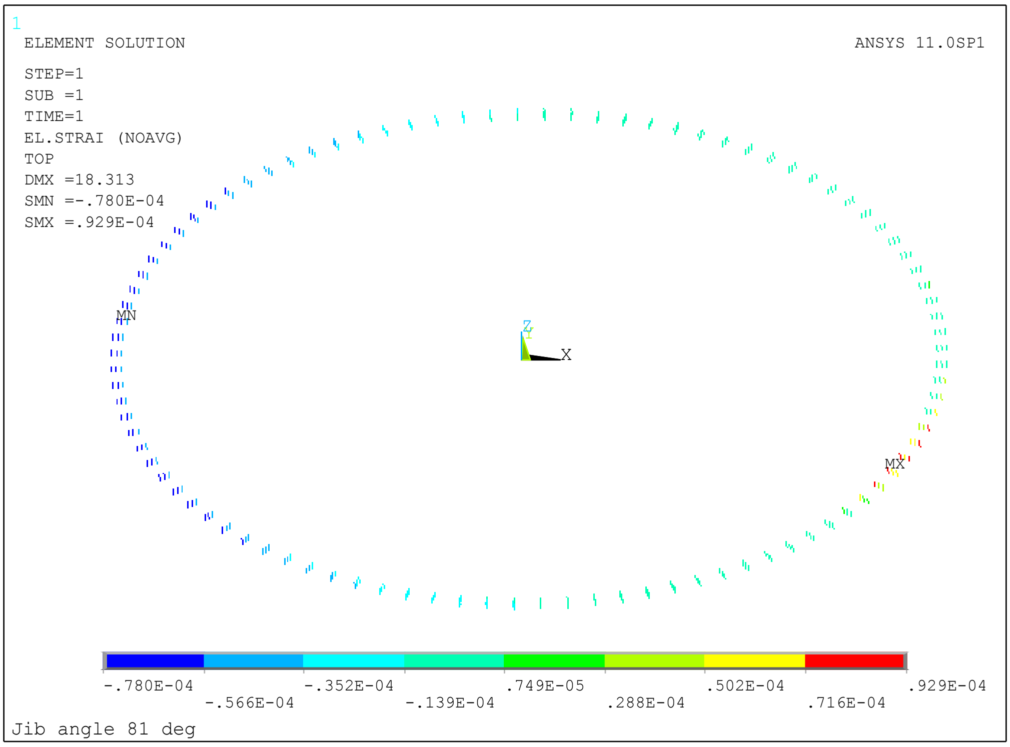

Figure 3.17: Elastic strain in bottom rolling elements

Radial gap

Since with growing diameters, the ratio between cross-sectional area and diameter is getting smaller, tangential elongation occurs and leads to a radial gap in the unloaded zone. The gap may be also recognized in elastic strain results (fig.3.16) though it is summed up in the following figures 3.18 and 3.19.

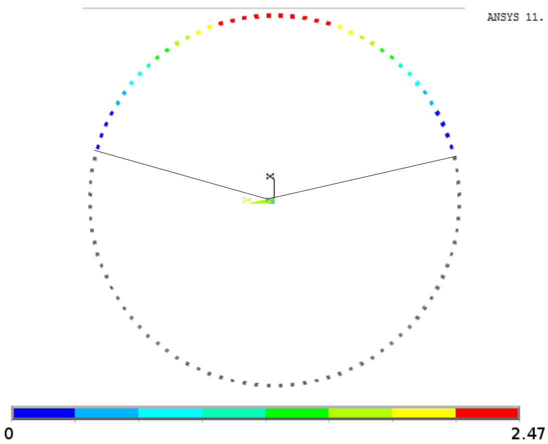

Figure 3.18: gap between radial rolling elements and the bearing (radial gap)

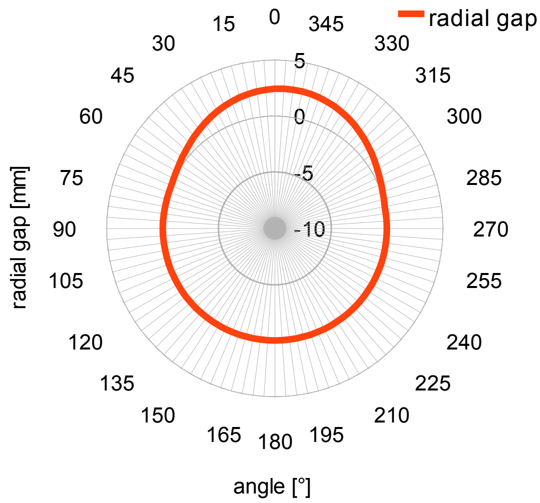

Figure 3.19: radial gap visualization

3.3.2. Expansion pass results

Flange and bearing stress distribution

Equivalent stress distribution (HMH) over the bearing and flange. Some elements below nodes, that simulate rolling elements, were omitted due to inappropriate high stress.

Only super-elements in location no.**1** (fig.3.20) has been postprocessed in expansion pass, both upper and bottom part of the bearing.

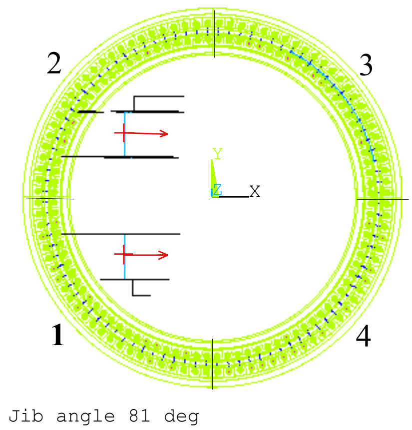

Figure 3.20: numbered SEs locations and applied forces

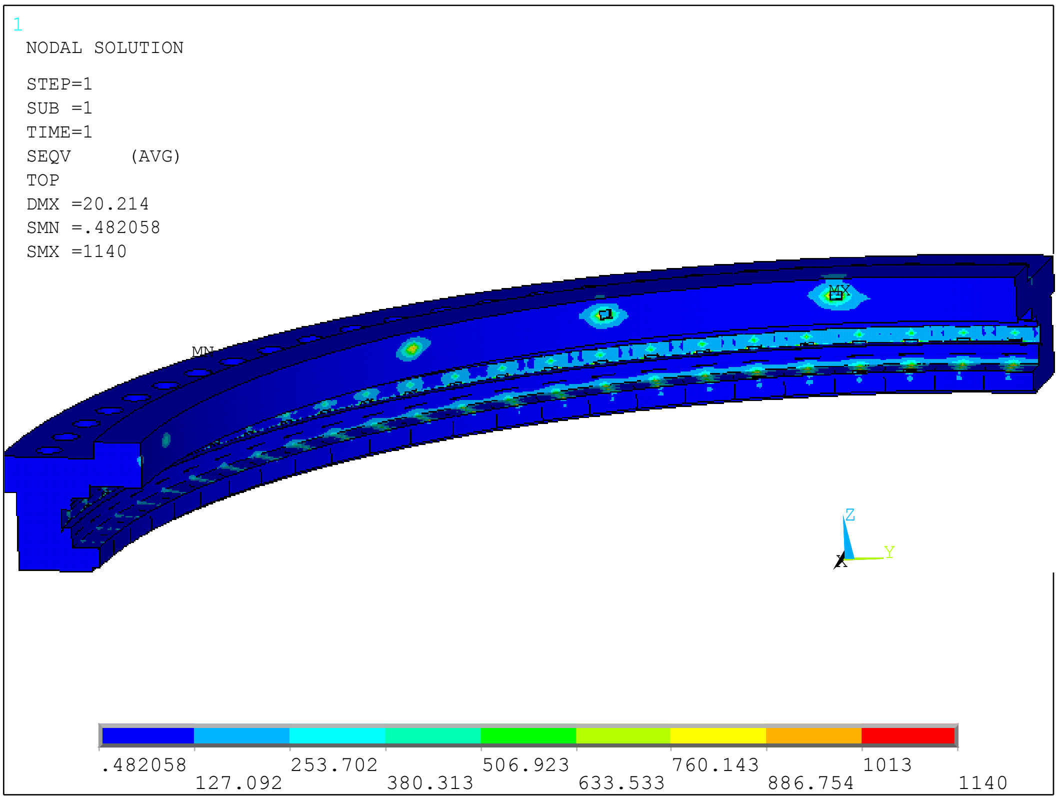

Figure 3.21: upper part of the bearing SE

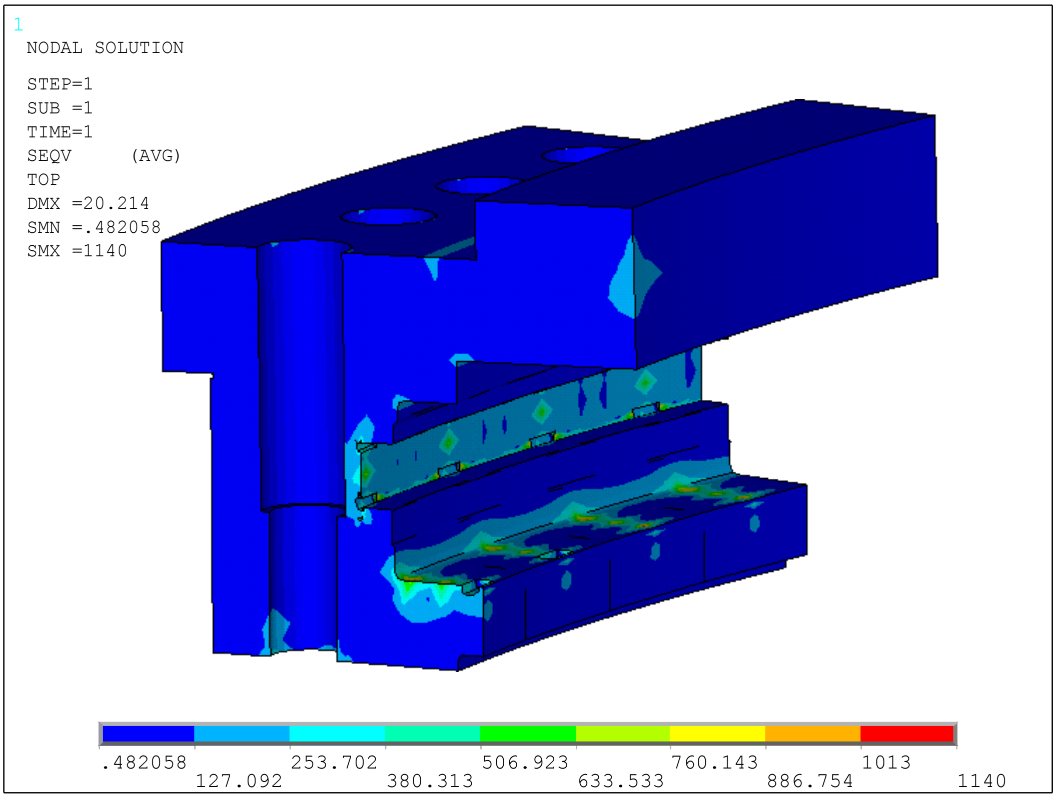

Figure 3.22: upper part of the bearing SE - detail of most stressed area

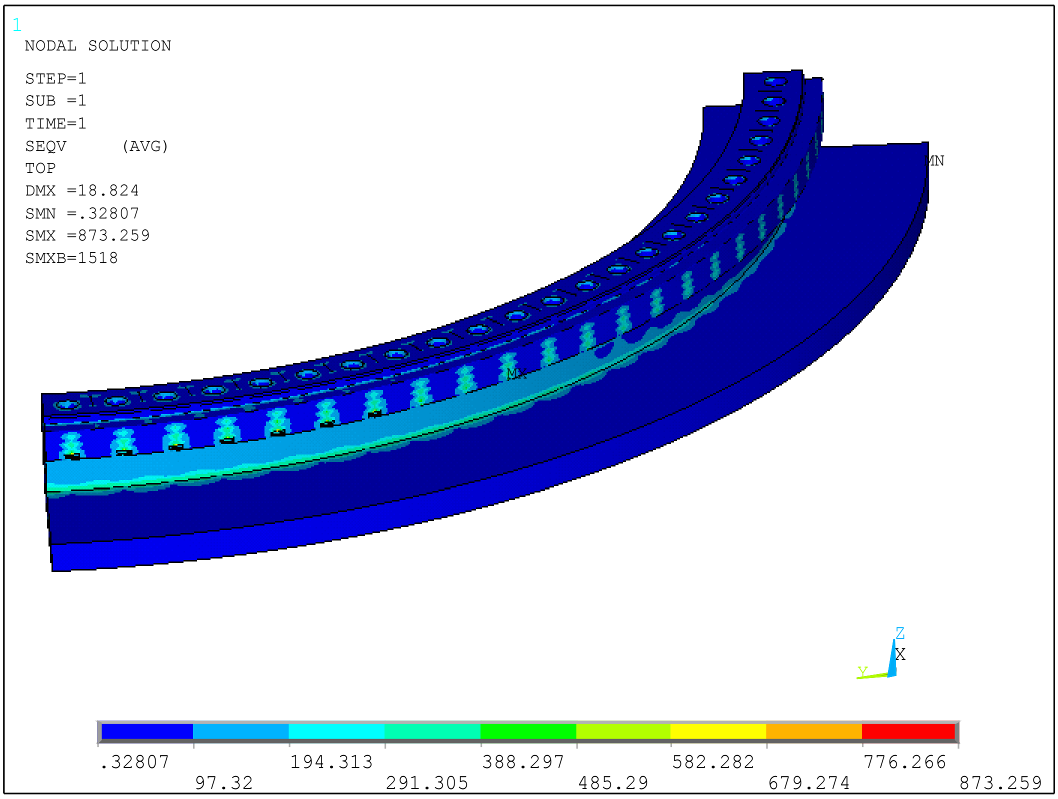

Figure 3.23: bottom part of the bearing SE

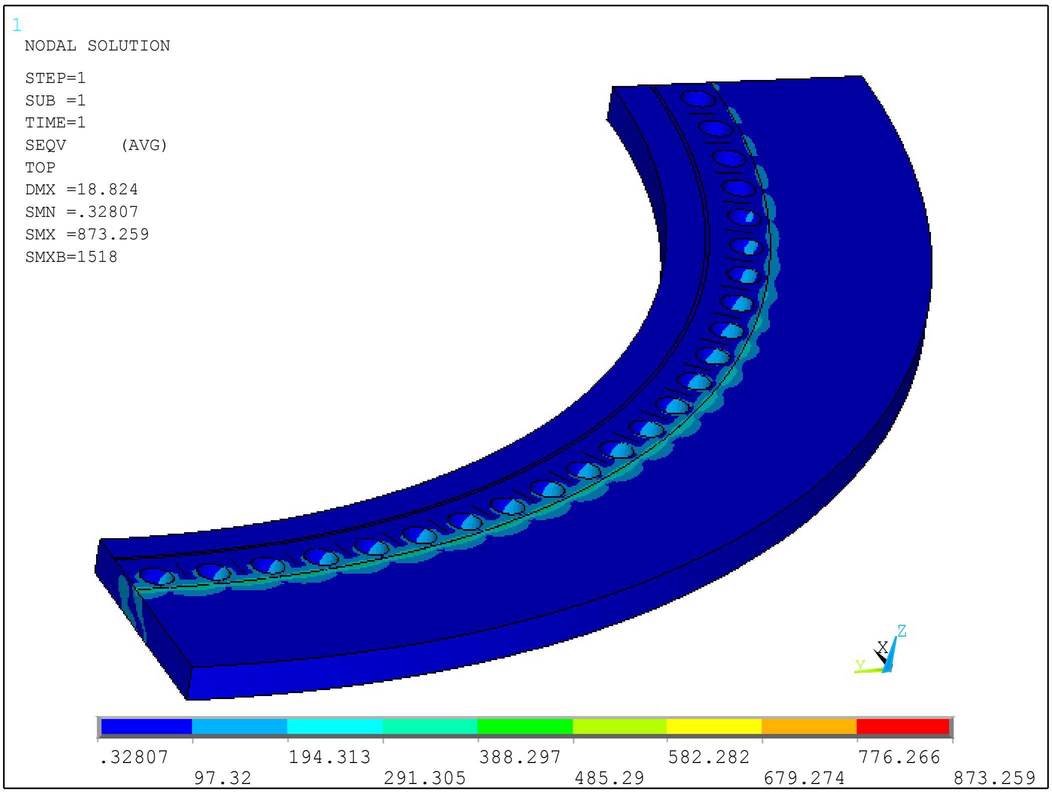

Figure 3.24: bottom part of the bearing SE - flange only

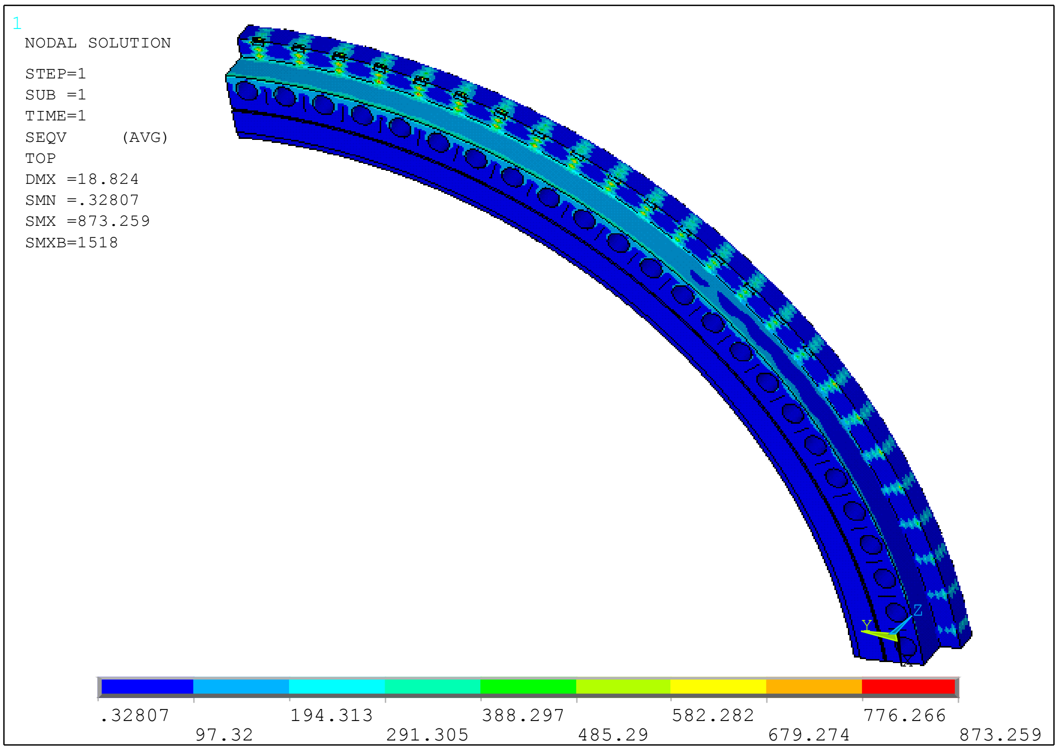

Figure 3.25: bottom part of the bearing SE - bearing only, view from below

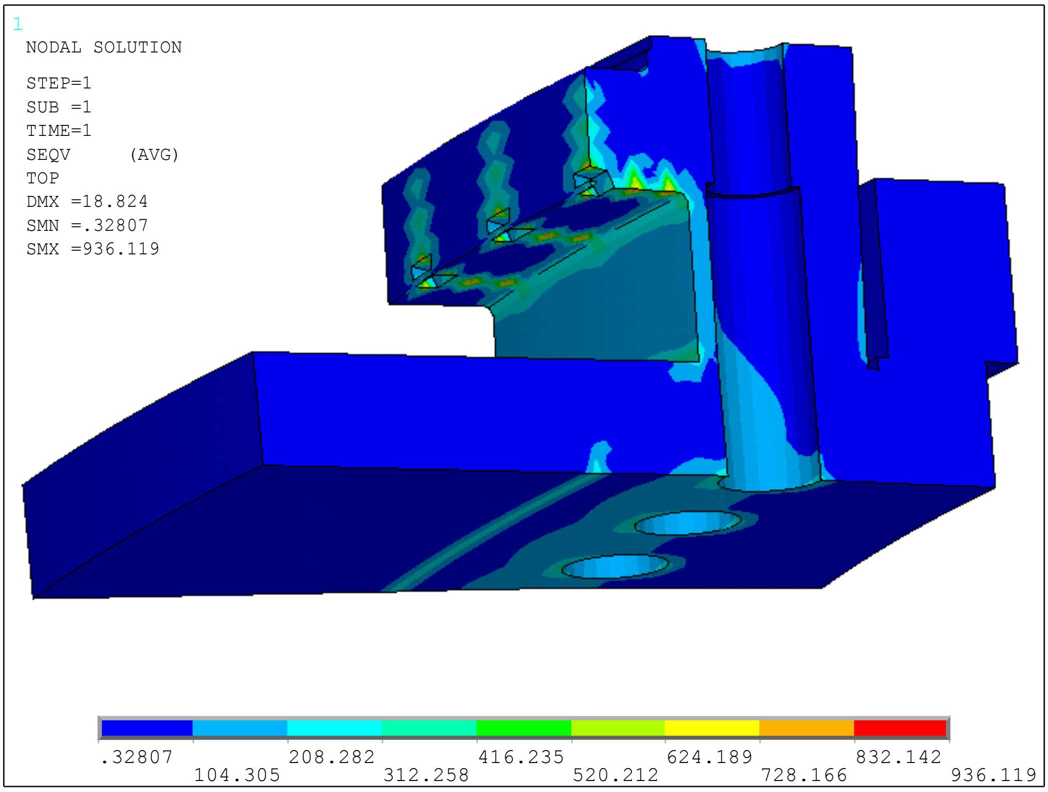

Figure 3.26: bottom part of the bearing SE - detail of most stressed area

4. Smart guide

This chapter summarize essential experience gained during application of SE technique in chapter 3.

4.1. Major issues

It is most important to pay attention to geometry and FE model with regard to its *Use Pass* integration from the very beginning.

Choosing SE proportion

Suggested SE proportion is 90 degree bearing segment. In figure 4.1 the division is pointed as numbers *1 – 4*. In the figure one may see that vertical stiffeners are connected with upper part of bearing (in red). The only symmetry for this stiffener

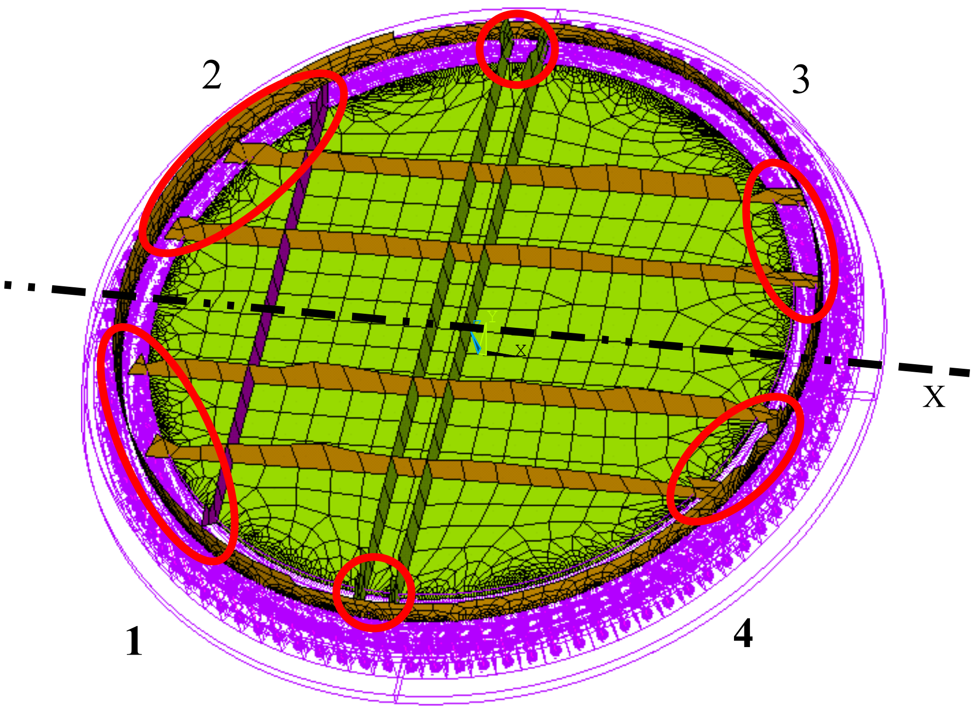

Figure 4.1: use pass model, upper bearing (SE) and surroundings; areas with non-symmetric master nodes pointed (red)

interfaces is X axis. Therefore half of the bearing has to be generated, the other half will be SE created by reflection. The unique half can be divided into two SEs. Depending on size of the model, generating larger chunk may lead in *generation pass* to fail.

Good mesh

Pay attention to make well regular mesh with as few DOFs as possible. Sweep meshing is very useful. It is possible to use only sweep although area around the hole has to be swept in different direction.

SE may be created by modelling one hole segment first. Afterwards the segment is copied. Finally master nodes are selected and SE is ready to be generated.

4.2. What to avoid

Too many **master** DOFs significantly increases requirements of hardware resources (more than a very few tens of thousands for use pass) and may lead to failure.

Limit DOF number

Number of DOF influences hardware resources that is needed for successful solution (generation pass in particular). One should avoid using elements with mid-nodes and irregular mesh.

Using **hard points** to get master nodes exactly where are needed is not recommended. If hard points are used in the bearing segment then it is not possible to mesh by sweep method. That would lead to worse controlled mesh and much larger amount of nodes which is highly important to keep at low rate.

5. Conclusion

Application of substructuring in masthead slewbearing analysis was successfully used.

Proper selection of SE proportion is major step in the analysis. Although one hole segment SE is the only unique geometry necessary in the bearing model itself, it is not unique when it comes to master node requirements (see chapter 4.). There is also an issue in realization of such approach in ANSYS. Additionally *use pass* model with such SEs would have too many master nodes due to very artificial model division. That would lead in longer computational time. Another extreme – one SE per bearing – has been tested as well but failed since the model was too large for generation pass.

Dividing the slewbearing in four SEs ensures that *use pass* model is properly connected still symmetry takes its place and saves time in generation pass process.

Depending on the model, above standard computers might be necessary for analysis. For solution in this thesis computers with large amount of memory, up to 16GB, were used in *generation* and *use pass*.

Results of the analysis in chapter 3 should be considered only as illustrative, proving that the analysis approach works. The thesis topic was too wide to embody all the provided technical details. Due to lack of time some minor issues were neglected leading in slightly inaccurate results. Yet none of those issues would affect the approach functionality.

Further improvement and testing could be done in

- better rolling element simulation – to eliminate stress peaks below the elements
- solid SE–shell joining – maybe SWGEN command would do it
- consideration of top-down substructuring approach

6. Sources

References

[1] *ANSYS – Substructuring*, Swanson Analysis System Inc., Houston 1992

[2] *ANSYS 11.0 Documentation*, SAS IP Inc., 2007

[3] PODEŠVA, J. *Dynamika rozsáhlých konstrukčních soustav, řešená metodou konečných prvků, redukce technikou rozkladu na substruktury.* Doktorská disertační práce. VŠB-TUO, 1999

[4] ŠEDĚNKA, M. *Dynamika mechanických soustav – technika rozkladu na substruktury*. Diplomová práce. VŠB-TUOO, 2005

[5] PRZEMIENIECKI, J.S. *Theory of Matrix Structural Analysis*. McGraw-Hill, New York, 1968

[6] *Introduction to 1800t slewbearing analysis*, Huisman – Itrec, Netherlands 2008

[7] *1800t HLMC Upperbearing Design report*, Huisman – Itrec, Netherlands 2008

[8] *Introduction to Slewbearings for Graduation Study*, Huisman – Itrec, Netherlands 2009

[9] Carlos A. Felippa, *INTRODUCTION to FINITE ELEMENT METHODS*, University of Colorado 2004

[10] *The Focus (Issue 45)*, a periodic electronic publication published by PADT, http://www.padtinc.com/epubs/focus/

[11] *Graduation-, Literature Study-, and Trainee Subjects*, Huisman – Itrec, Netherlands 2007

Software

[1] ANSYS release 11

[2] Open Office – office suite (open source)

[3] Gimp – raster graphics editor (open source)

[4] Inkscape – vector graphics editor (open source)